SIMULATION AND FLOW ANALYSIS THROUGH A STRAIGHT PIPE

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF
THE REQUIREMENTS FOR THE DEGREE OF

Bachelor of Technology
In
Civil Engineering

By:

SASWAT SAMBIT
110CE0056

UNDER THE GUIDANCE OF:

Prof. AWADHESH KUMAR

DEPARTMENT OF CIVIL ENGINEERING
NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA

CERTIFICATE

This is to certify that the project report entitled "SIMULATION AND FLOW ANALYSIS THROUGH A STRAIGHT PIPE" submitted by Mr. Saswat Sambit in partial fulfillment of the requirements for the award of Bachelor of Technology Degree in Civil Engineering at National Institute of Technology, Rourkela (Deemed University) is an authentic work carried out by them under our supervision and guidance.

To the best of our knowledge, the matter embodied in this Project Report has not been submitted to any other University/Institute for the award of any Degree or Diploma.

Date :

Prof. Awadhesh Kumar

ACKNOWLEDGEMENT

The satisfaction on the successful completion of any task would be incomplete without the mention of the people associated with it, whose constant guidance and encouragement gave shape to the vision of this project.

I am immensely grateful to the Department of Civil Engineering, NIT Rourkela for providing me with an opportunity to accomplish this project.

I would like to express the deepest appreciation to my project guide, Prof. Awadhesh Kumar, Department of Civil Engineering, for his constant support, inspiration and keen interest that that played an integral part in the completion of this project.

My special thanks Prof. N Roy, Head of Department, Civil Engineering for all the facilities provided to successfully complete this work. I am also grateful to all faculty members of the department, especially Water Resources specialization for their constant encouragement, invaluable advice, encouragement and inspiration and blessings during the project.

Lastly, I express my gratitude to all who have been directly or indirectly involved in this project.

Saswat Sambit

110CE0056

CONTENTS

LIST OF FIGURES

LIST OF TABLES

ABSTRACT

In this study, the frictional losses in a pipe due to shear stress produced in a pipe due to viscosity of fluids has been discussed. Pipeline system in India is a vast network and minimizing losses through these pipes can contribute a lot to the country. This paper contains flow analysis of different fluids in a pipe.

The simulations were done using ANSYS FLUENT CFD 14.0 software to observe pressure drops between inlet and outlet of a pipe. Pressure Difference and frictional coefficient were calculated using Darcy-Weisbach equation. The model used in ANSYS FLUENT CFD 14.0 is a 3 dimensional model through which different fluids were made to pass through and subsequent analysis were done. The paper also includes comparison of results obtained from practical methods, theoretical methods and ANSYS. The flow is considered to be turbulent and open channel flow.

Keywords: frictional loss; frictional coefficient; viscous fluid; velocity profile

CHAPTER 1

INTRODUCTION

1.1 GENERAL

Pipe network is very common in industries throughout the country, where fluid and gases are transported from one point to another. The pressure loss depends on the type of flow of the fluid in the network, pipe material, and the fluid flowing through the pipe. When any fluid flows through a pipe, the velocity adjacent to the pipe wall is zero and the velocity gradually increases from the wall. Maximum velocity is observed at the centre of the pipe. Due to increase in the velocity gradient, shear stresses are produced in the fluid due to its viscosity. This viscous action attributes to loss of energy which is commonly known as loss due friction or frictional loss.

William Froude stated the following laws of fluid fraction under turbulent flow.

For a turbulent flow, frictional resistance is:

1. Directly proportional to V^n, where n varies between 1.5 to 2.
2. Proportional to fluid density
3. Proportional to surface area in contact.
4. Independent of the pressure
5. Dependent on the nature of the surface in contact.

If losses are minimum in a pipe network then the efficiency is higher. Moreover, all networks should be designed to undergo minimum loss.

1.2 OBJECTIVE AND SCOPE

The objective of the study is to compare frictional losses in a standard pipe using different pipe materials and fluids flowing through pipe. Following points represent the scope for this study.

- All flows are considered to be turbulent.
- Flow through the pipe is considered to be open channel flow.
- The pipe material used is Brass, Galvanized Iron and Stainless Steel.
- The fluids used are water, liquid ammonia, crude oil and diesel.
- The basis of choosing the fluids was on their importance in industries and popularity to be transported through pipe networks.

1.3 METHODOLOGY

This project can broadly divided into the following stages.

1) Identifying the problem statement and formulating objectives.
2) Preparation for project:
    - This includes all preparatory things like background study, literature review, data collection from laboratory practical etc.
    - Laboratory practical that are to be undertaken for this project are frictional losses in pipe a straight pipe.
    - Various models of pipes are to be modelled in ANSYS Software for the analysis and comparison of the results from laboratory and Ansys.
3) Optimization of result:
    - Flow analysis for different fluids flowing through different pipe materials using data obtained from practical, theoretical and ANSYS methods.
    - Comparison of results with different diameter of pipe.

CHAPTER 2

LITERATURE REVIEW

LITERATURE REVIEW

1. Abdulwahhab, N Kumar and Fahad conducted an extensive 3D numerical parametric investigation of turbulent flow in 90° T junction and published journals (International Journal of Engineering Science and Technology) in 2012.

2. Sierra-Espinoza and Bates used various turbulence models and concluded that RNG and RSM turbulence models predicted the mean flow quantitatively in 2000.

3. Miller stated that the major losses for T and Y junctions are due to combination and division of flows, which arises from separation and subsequent turbulent mixing.

CHAPTER 3

STUDY AREA

3.1 ENERGY LOSSES IN PIPE

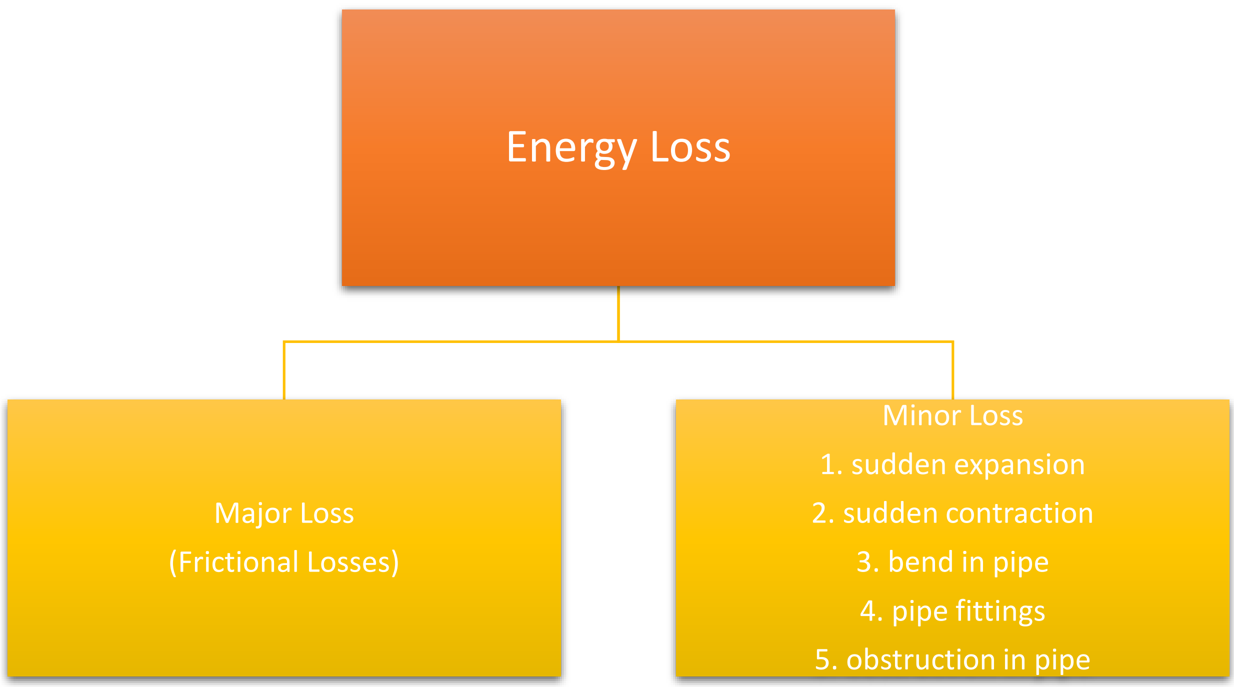
When a fluid is flowing through a pipe, the fluid experiences resistance due to which it loses some energy. This energy loss can be classified as:

1. Major Losses

   This is due to friction and is also termed as frictional loss.

2. Minor Losses

   This is due to sudden expansion, contraction, pipe fittings, bend in pipe, obstruction in pipe etc.



3.1.1 MAJOR LOSSES OR FRICTION LOSSES

Friction loss is the loss of energy or "head" that occurs in pipe flow due to viscous effects generated by the surface of the pipe. Friction Loss is considered as a "major loss" and it is not to be confused with "minor loss" which includes energy lost due to obstructions. In mechanical systems such as internal combustion engines, it refers to the power lost overcoming the friction between two moving surfaces.

This energy drop is dependent on the wall shear stress (τ) between the fluid and pipe surface. The shear stress of a flow is also dependent on whether the flow is turbulent or laminar. For turbulent flow, the pressure drop is dependent on the roughness of the surface, while in laminar flow, the roughness effects of the wall are negligible. This is due to the fact that in turbulent flow, a thin viscous layer is formed near the pipe surface which causes a loss in energy, while in laminar flow, this viscous layer is non-existent.

Friction loss has several causes, including:

- Frictional losses depend on the conditions of flow and the physical properties of the system.
- Movement of fluid molecules against each other.
- Movement of fluid molecules against the inside surface of a pipe or the like, particularly if the inside surface is rough, textured, or otherwise not smooth.
- Bends, kinks, and other sharp turns in hose or piping.

In pipe flows the losses due to friction are of two kinds: skin-friction and form-friction. The former is due to the roughness of the inner part of the pipe where the fluid comes in contact with the pipe material, while the latter is due to obstructions present in the line of flow--perhaps a bend, control valve, or anything that changes the course of motion of the flowing fluid.

1. DARCY-WEISBACH FORMULA

One of the accepted methods to calculate friction losses resulting from fluid motion in pipes is by using the Darcy-Weisbach Equation. For a circular pipe:

$$h_i = \frac{fLV^2}{2gD}$$

where: h_i = Head Loss due to friction, given in units of length

f = friction factor (Darcy-Weisbach friction coefficient)

L = Pipe Length

h_i = Head Loss due to friction, given in units of length

f = friction factor (Darcy-Weisbach friction coefficient)

L = Pipe Length

D = Pipe Diameter

V = Flow velocity

g = acceleration due to gravity

2. CHEZY'S FORMULA

$$V = C\sqrt{mi}$$

V- mean velocity of flow

C – Chezy's constant

m – hydraulic mean depth

3.1.2 MINOR LOSSES

The additional components such as valves and bend add to the overall head loss of the system, which is turn alters the losses associated with the flow through the valves.

Minor losses termed as;

$h = K \left(\frac{V^2}{2g}\right)$

where K is the loss coefficient.

Each geometry of pipe entrance has an associated loss coefficient.

CHAPTER 4

TEST APPARATUS

AND

PROCEDURE

4.1. MAJOR LOSS TEST APPARATUS

The apparatus consist of three pipes with G.I pipe, Brass pipe, Stainless Steel pipe, all of 12.7 mm diameter, so that loss of head can be compared for different materials. A flow control valve is provided at outlet of pipes which enables experiments to be conducted at different flow rates, i.e. at different velocities.

Tapings are provided along the length of pipes, so that drop of head can be visualized along the length of pipe. Each pipe is provided with valve at outlet, which enables heads to be controlled.

Fig 4.a. FRICTION LOSS TEST SETUP

4.2 EXPERIMENTAL PROCEDURE

- Water in the sump tank was filled. (This water should be free of any oil content.)
- All the outlet valves were opened and the pump started.
- By closing all of outlet valves, for each pipe leakages were checked, and correction was made for the leaks, if any.
- All the outlet valves of the pipe to be tested were opened.
- All the air bubbles from manometer and connecting pipes were removed.
- The flow was reduced. Outlet valves adjusted, so that water heads in manometer are to the readable height.
- The heads and flow rate were observed.
- Now, the flow was increased and accordingly the outlet valve was adjusted, so that water will not overflow.
- The procedure for were repeated for other pipes.

4.3 EXPERIMENTAL OBSERVATION

1. G.I. PIPE

Length of pipe = 1m

Diameter of pipe = 12.7 mm

TABLE 4.1 (OBSERVATION FOR GI PIPE)

Sl No	Manometer Reading (mm)	Head (mm)	Flow Rate (sec)	Discharge (m^3/sec)	Velocity (m/sec)	Frictional coefficient
1	55	693	70.9	3.24×10^{-4}	2.56	0.0264
2	119	1499.4	45.84	5.02×10^{-4}	3.96	0.024
3	145	1827	39.32	5.84×10^{-4}	4.61	0.0216

2. BRASS PIPE

Length of pipe = 1m

Diameter of pipe = 12.7 mm

TABLE 4.2 (OBSERVATION FOR A BRASS PIPE)

Sl No	Manometer Reading (mm)	Head (mm)	Flow Rate (sec)	Discharge (m^3/sec)	Velocity (m/sec)	Frictional coefficient
1	59	743.4	66.6	3.45×10^{-4}	2.42	0.024
2	115	1449	47.7	4.82×10^{-4}	3.8	0.024
3	147	1852.2	39.3	5.85×10^{-4}	4.62	0.022

3. STAINLESS STEEL PIPE

Length of pipe = 1m

Diameter of pipe = 12.7 mm

TABLE 4.3 (OBSERVATION FOR A STAINLESS STEEL PIPE)

Sl No	Manometer Reading (mm)	Head (mm)	Flow Rate (sec)	Discharge (m^3/sec)	Velocity (m/sec)	Frictional coefficient
1	61	768.6	64.64	3.56×10^{-4}	2.81	0.0244
2	103	1297.8	44.56	5.12×10^{-4}	4.04	0.02
3	125	1575	39	5.88×10^{-4}	4.65	0.0184

Friction coefficient for GI pipe = 0.024

Friction coefficient for Brass pipe = 0.023

Friction coefficient for Stainless Steel pipe = 0.02

4.4 MOODY DIAGRAM

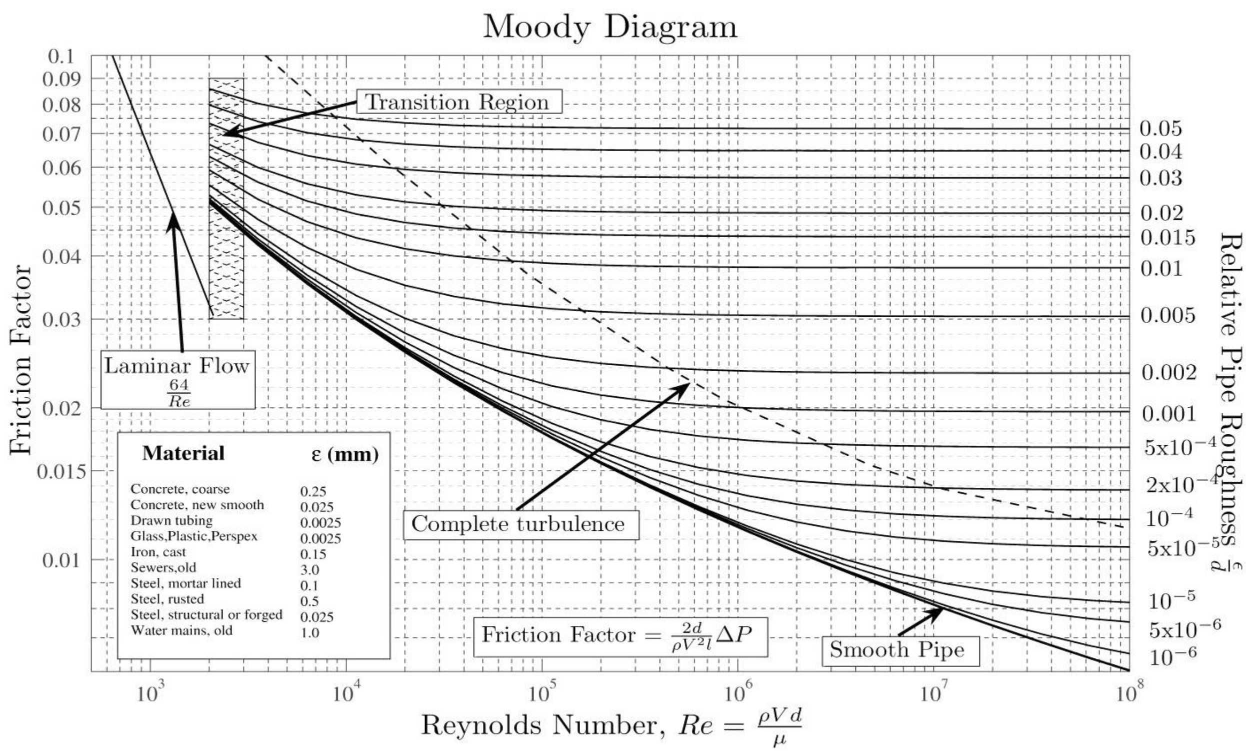


Fig 4.b. MOODY DIAGRAM

- The Moody diagram helps in determining the coefficient of friction of a pipe. The factor can be determined by its Reynolds number and the Relative roughness of the Pipe.
- The rougher the pipe the more turbulent the flow is through that pipe.
- The relative roughness of a pipe is given by

  $\frac{e}{D}$ where, e – absolute roughness and D – dia of pipe
- Reynold's number is given by

$$R = \frac{Dv}{\zeta}$$

Where: R = Reynolds number

D = diameter

v = velocity

ζ = kinematic viscosity of fluid

Following tables denote the values of kinematic viscosity and absolute roughness of fluids and pipe materials respectively. Data given are predetermined values and are taken from reference books.

TABLE 4.4 (ABSOLUTE ROUGNESS OF PIPE MATERIALS)

MATERIALS	ABSOLUTE ROUGHNESS (mm)
Galvanized Iron	0.015
Stainless Steel	0.015
Brass	0.0015

TABLE 4.5 (KINEMATIC VISCOSITY OF WATER AT DIFFERENT TEMPERATURE)

TEMPERATURE (˚C)	KINEMATIC VISCOSITY (m²/sec)× $\mathbf{10^{-6}}$
0	1.787
5	1.519
10	1.307
20	1.004
30	0.801
40	0.658
50	0.553

At 25˚ C, kinematic viscosity = 0.9025

4.5 OBESRVATION FROM MOODY DIAGRAM

Following is a tabular form of the observation obtained from MOODY DAIGRAM. From the diagram the values of frictional coefficient were obtained with respect to the values of Reynold's number and relative pipe roughness.

TABLE 4.6 (OBSERVATION FROM MOODY DIAGRAM)

MATERIAL	REYNOLD'S NUMBER	RELATIVE ROUGHNESS	FRICTION COEFFICIENT
GALVANIZED IRON	36024	0.0012	0.027
GALVANIZED IRON	55725	0.0012	0.025
GALVANIZED IRON	64872	0.0012	0.024
BRASS	34054	0.0001	0.026
BRASS	53473	0.0001	0.024
BRASS	65012	0.0001	0.023
STAINLESS STEEL	39542	0.0012	0.024
STAINLESS STEEL	56850	0.0012	0.023
STAINLESS STEEL	65434	0.0012	0.022

Friction coefficient for GI pipe = 0.025

Friction coefficient for Brass pipe = 0.024

Friction coefficient for Stainless Steel pipe = 0.022

CHAPTER 5
MODELLING
AND
FLOW ANALYSIS
USING ANSYS

5.1 ANSYS FLUENT 14.0

ANSYS, Inc. is an engineering simulation software (computer-aided engineering, or CAE) developer that uses CFD, FEM and other various programming algorithms for simulation and optimization.

In this study, ANSYS FLUENT was used to analyze the flow in pipes. This software follows 5 steps for completion of any project. They are as follows.

1. Modelling

First step is modelling of the material to be analyzed. In this study, a pipe of 12.7 mm diameter and 1m length is modelled. In spite of modelling the complete pipe, one quadrant of the circular pipe is modelled as the all the properties will be symmetry throughout the pipe. Below is the diagram of the model used in this study.

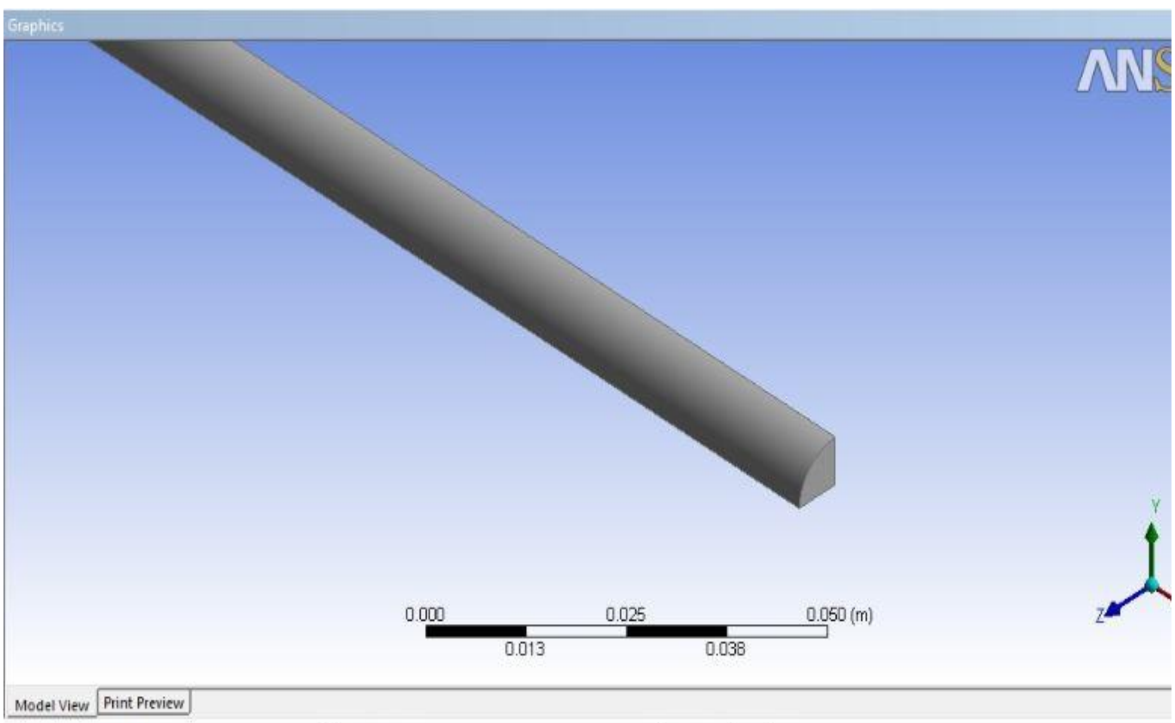


Fig 5.a MODEL OF THE PIPE

2. MESHING

The model is meshed to get the properties accurately. Meshing means diving the model in numerous smaller equivalent parts so that analysis becomes easier. Analysis is done for every meshed area and the summation of all the areas shows the total property gradient of the model. One can control the meshing by choosing different properties of mesh like size of mesh area, meshing style, mesh thickness etc. Following figure illustrates a meshed model of the pipe used.

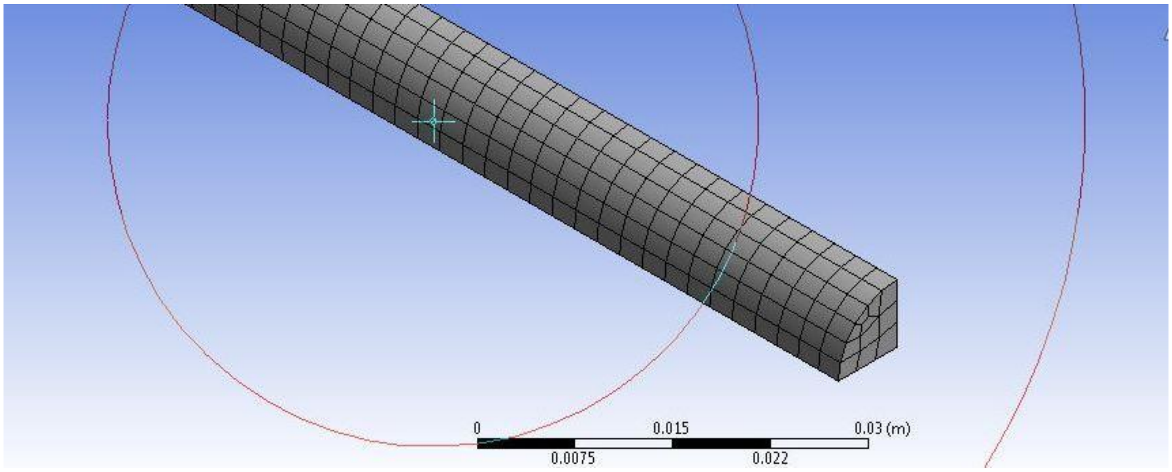


Fig 5.b MESHING OF THE MODEL

3. SOLUTION

This step involves feeding all prerequisite data such as pipe roughness, inlet velocity, type of flow, type of fluid, flow percentage, initializing the flow, number of iterations etc. Once the data are provided, calculation is done and immediately the iteration begins. When all the points converge, the calculation stops.

4. RESULT

After solution step, result step includes obtaining the results like pressure difference at inlet and outlet, net pressure in the pipe, outlet velocity etc. These results are used to calculate loss in the pipe and subsequently friction coefficient.

5. REPORT

This is the final step of a project in ANSYS which helps in displaying the property gradients of the model such as velocity, total pressure, static pressure, shear, eddy viscosity in form of contours, streamline or particle motion form. Following is an example denoting the velocity gradient in an elbow bend pipe. Here, in example the model used in the study isn't being shown as the pipe is very large making it difficult for the viewer to see the complete gradients throughout the pipe clearly.

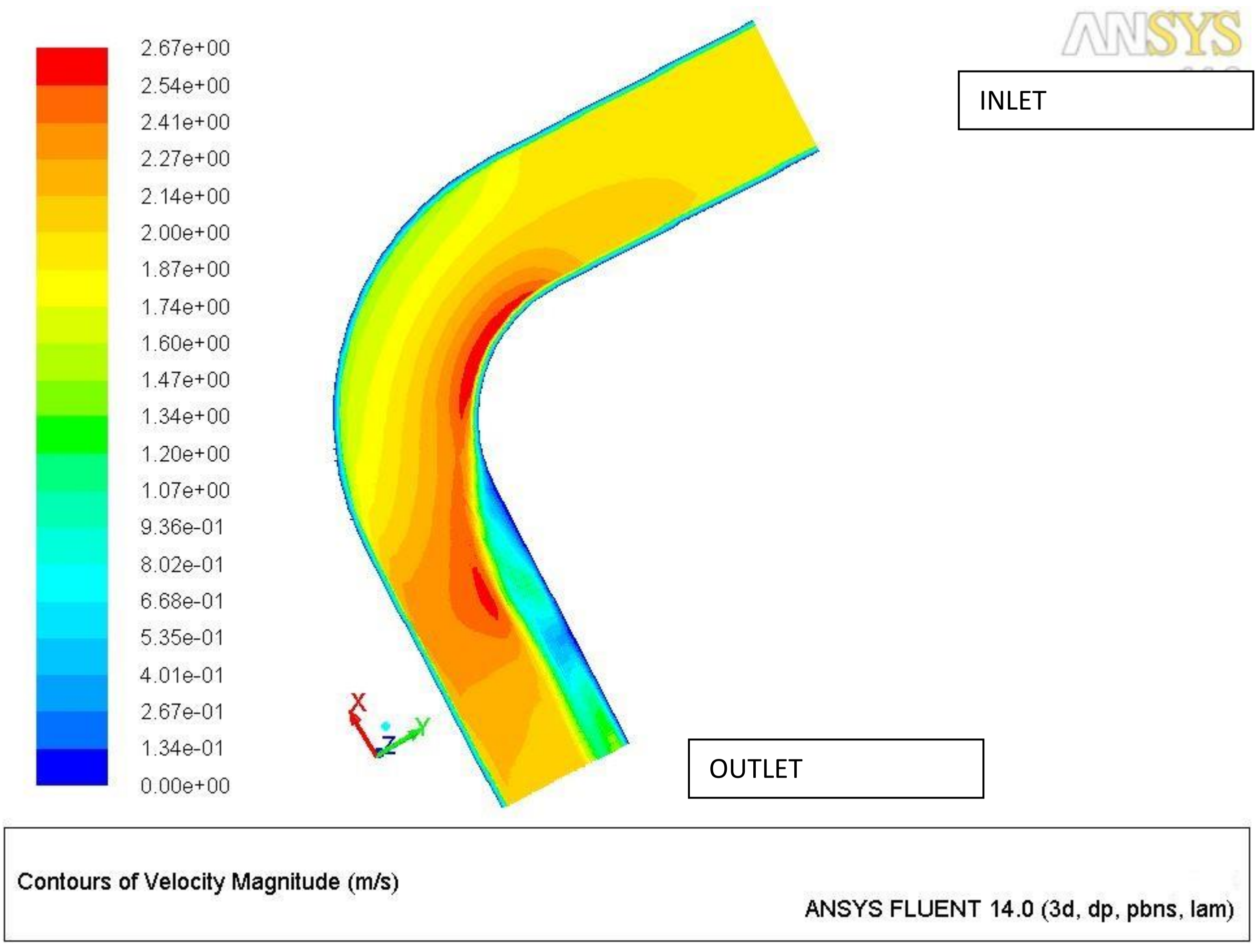


Fig 5.c VELOCITY CONTOUR OF AN ELBOW PIPE

5.2 CALCULATION OF LOSS USING ANSYS

- A model of 1m length and 12.7mm diameter is drawn. The model is one quadrant of the pipe.
- Since the study doesn't require calculating different properties at every point a general meshing of the model is done. Generated mesh is a controlled by the software itself which is considered suitable for this model.
- Name selection is done, i.e. inlet, outlet, pipe wall, symmetry faces (sym1 and sym2).
- The flow is taken to be turbulent, flow ratio 10%, number of iteration 500.
- Inlet velocity, pipe material, fluid type are set accordingly.
- After running calculation, iteration begins and convergences at a point.
- Pressure difference between the inlet and outlet of the pipe is shown by the software.
- Using Darcy Weisbach's equation, head loss and coefficient of friction is calculated.

5.3 OBSERVATION

TABLE 5.1 (OBSERVATION FROM ANSYS)

FLUID	MATERIAL	FRICTIONAL COEFFICIENT
WATER	GALVANIZED IRON	0.021
WATER	STAINLESS STEEL	0.02
WATER	BRASS	0.021
LIQUID AMMONIA	GALVANIZED IRON	0.0098
LIQUID AMMONIA	STAINLESS STEEL	0.0092
LIQUID AMMONIA	BRASS	0.0097
DIESEL	GALVANIZED IRON	0.028
DIESEL	STAINLESS STEEL	0.026
DIESEL	BRASS	0.0278
CRUDE OIL	GALVANIZED IRON	0.121
CRUDE OIL	STAINLESS STEEL	0.114
CRUDE OIL	BRASS	0.14

5.4 VELOCITY AND PRESSURE PROFILE

Following figures show the velocity and pressure gradients through the pipe. Fluids considered are water, liquid ammonia, diesel and crude oil and pipe materials are brass, galvanized iron and stainless steel.

5.4.1 G.I. PIPE

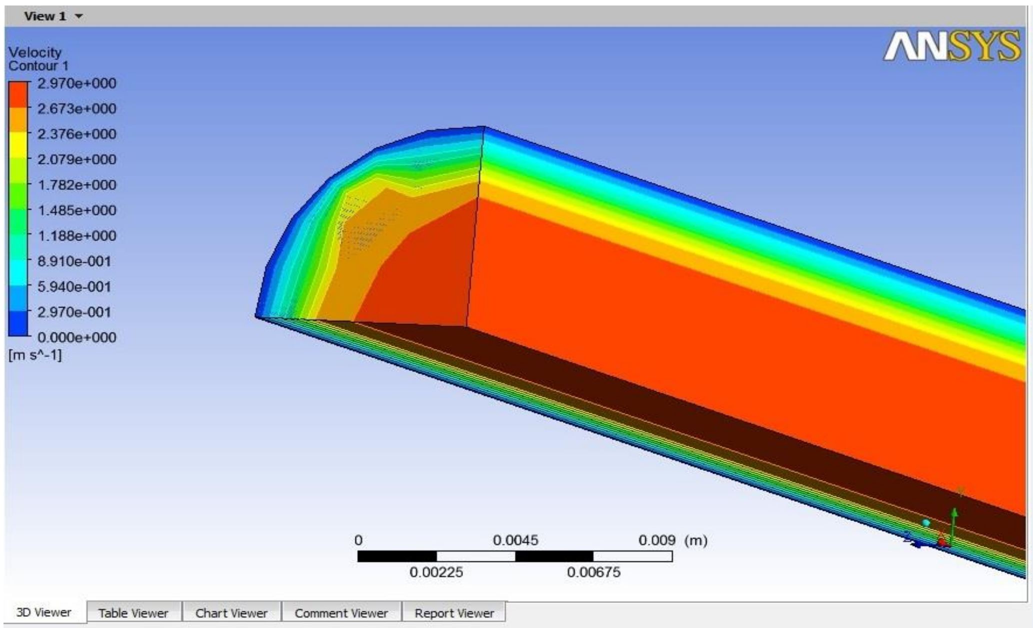


Fig 5.d. VELOCITY CONTOUR FOR WATER

The inlet velocity is 2.56m/sec and the fluid used is water. The figure shows that the velocity is maximum at the centre of the pipe. Maximum velocity observed is 2.97m/sec and minimum velocity 0 at the pipe wall.

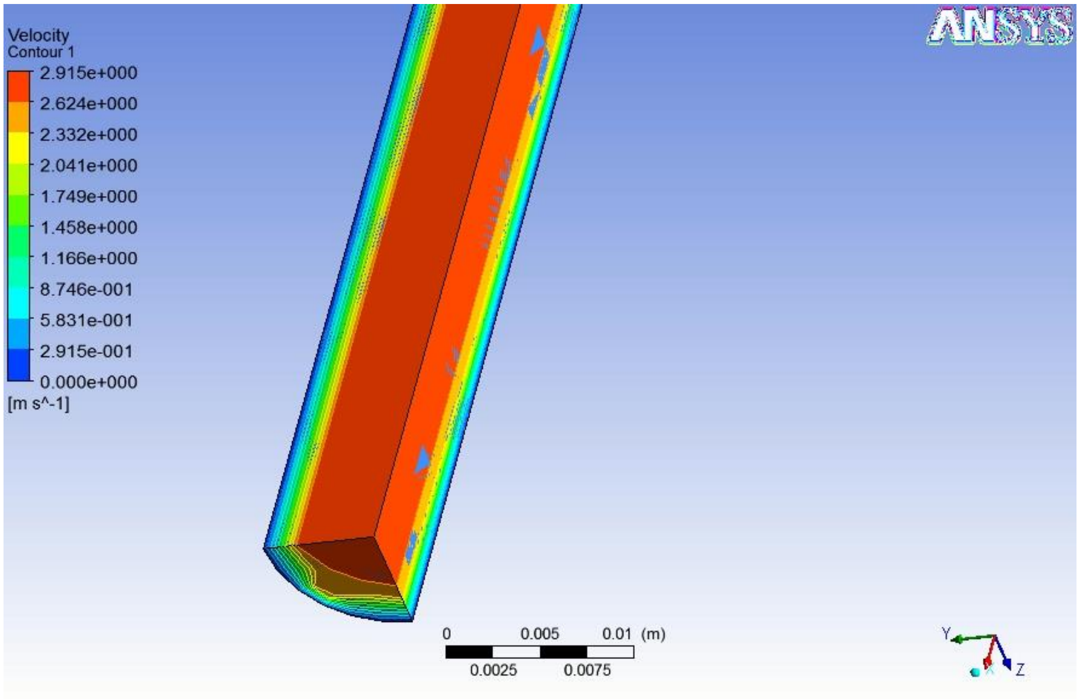


Fig 5.e VELOCITY CONTOUR FOR LIQUID AMMONIA

The inlet velocity is taken as 2.56 m/sec. The maximum velocity observed is 2.915 m/sec and the minimum velocity is 0 at the walls.

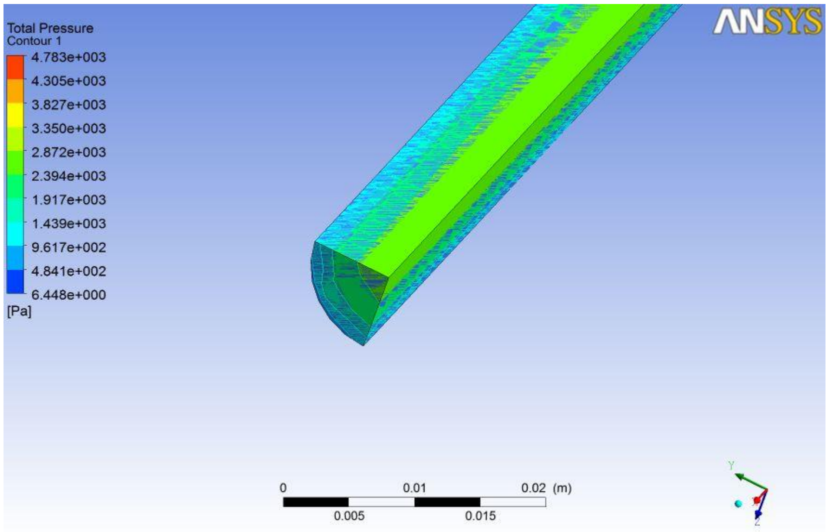


Fig 5.f TOTAL PRESSURE CONTOUR FOR LIQUID AMMONIA

Maximum pressure observed is 4.783 KPa and minimum is 6.446 Pa. Maximum pressure occurs at the inlet of the pipe.

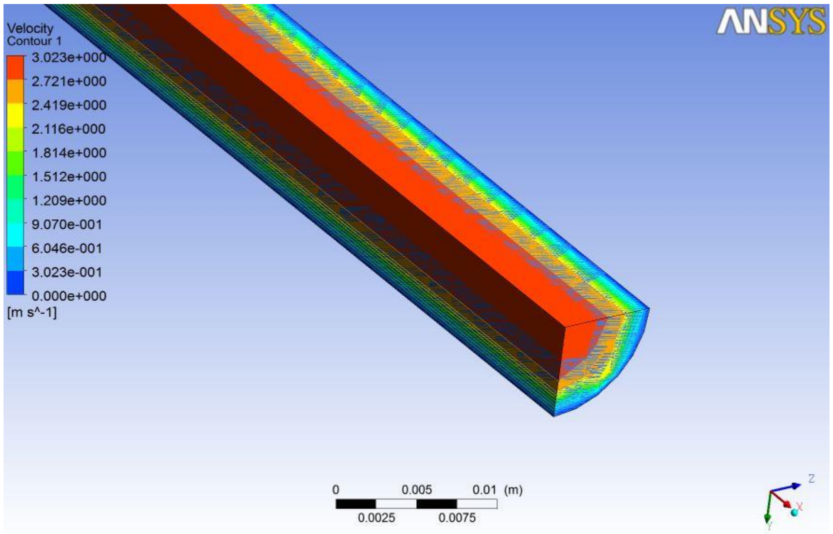


Fig 5.g. VELOCITY CONTOUR FOR DIESEL

Inlet velocity is 2.56 m/sec and head loss is much higher than that observed in liquid ammonia and water. Maximum velocity is 3.023 m/sec and minimum is 0 at the walls.

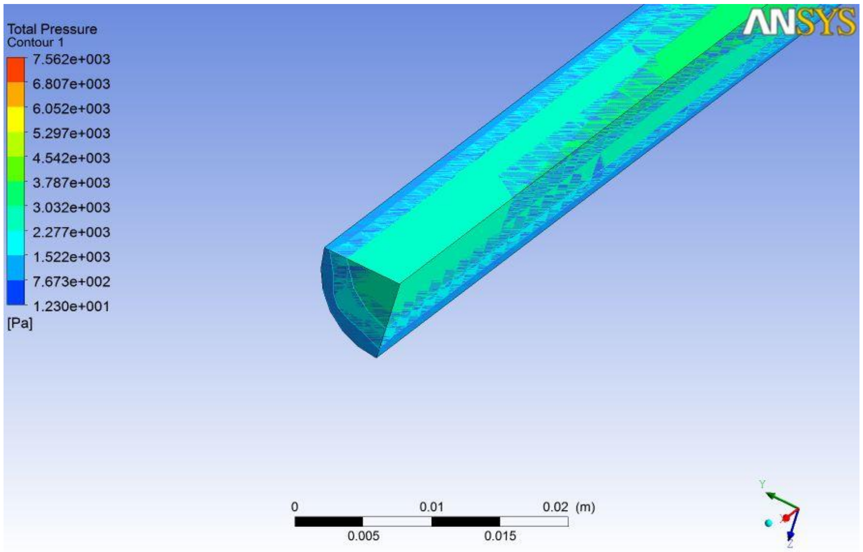


Fig 5.h. TOTAL PRESSURE CONTOUR FOR DIESEL

Total pressure contour illustrates that the pressure is very high with respect to that observed in liquid ammonia and water and thus diesel flow experiences more energy loss. Maximum pressure is 7.562 KPa and minimum is 1.23 Pa.

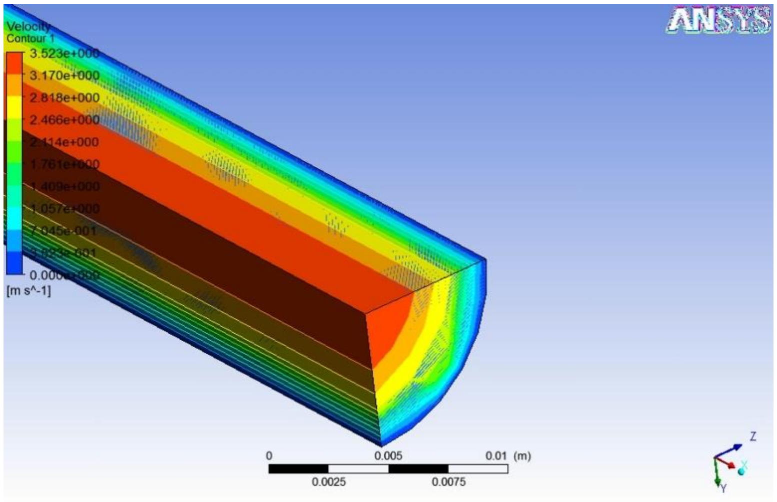


Fig 5.i. VELOCITY CONTOUR FOR CRUDE OIL

Maximum velocity observed in the pipe is 3.52 m/sec and minimum velocity 0 at the walls. Inlet velocity is taken as 2.56 m/sec.

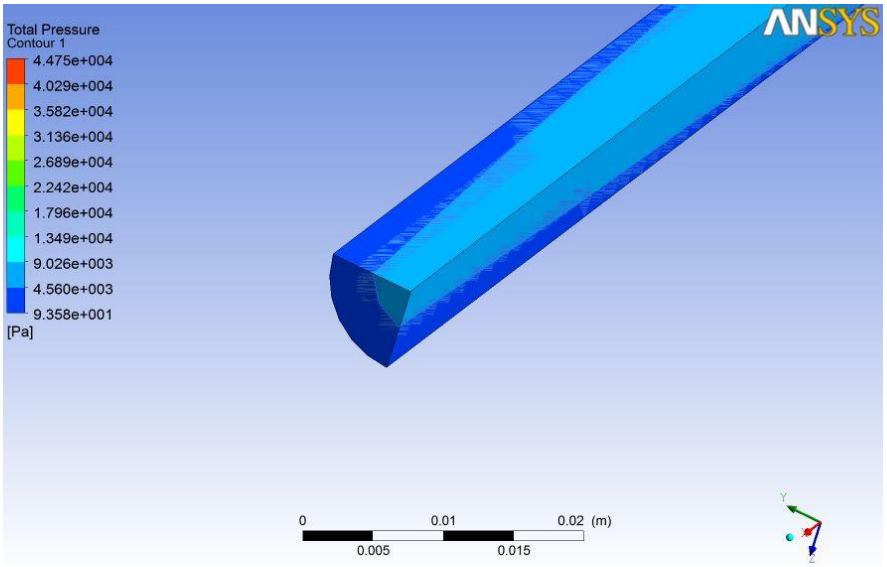


Fig 5.j. TOTAL PRESSURE CONTOUR FOR CRUDE OIL

Pressure exerted by crude oil is maximum ranging from 4.475 KPa to 9.35 Pa. This is due to high viscosity of crude oil. Energy loss is maximum for crude oil.

5.4.2 BRASS PIPE

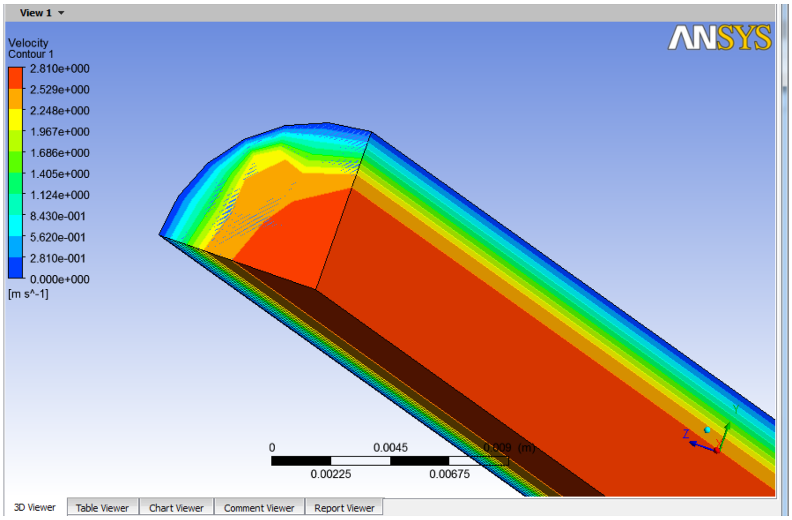


Fig 5.k. VELOCITY CONTOUR FOR WATER

Above figure shows the velocity profile for water with inlet velocity of 2.42 m/sec. Maximum velocity observed is 2.81 m/sec and minimum is 0 at the walls. Loss of energy incase of Brass pipe is more than compared to G.I Pipe.

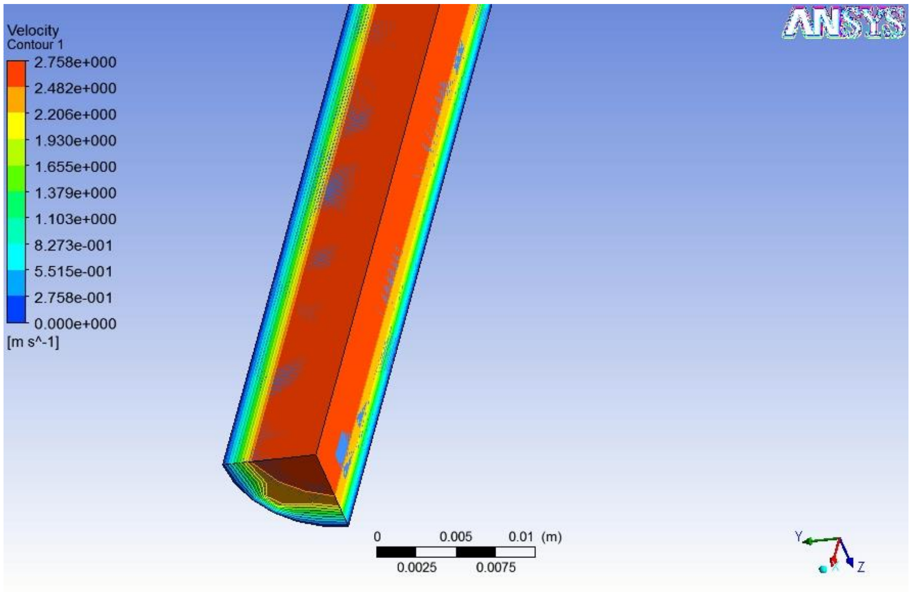


Fig 5.k.i. VELOCITY CONTOUR FOR LIQUID AMMONIA

Ammonia experiences lesser maximum velocity than water as it has low viscosity. Maximum and minimum velocities range from 2.75m/sec to 0 respectively with maximum velocity at the centre and minimum velocity at the wall.

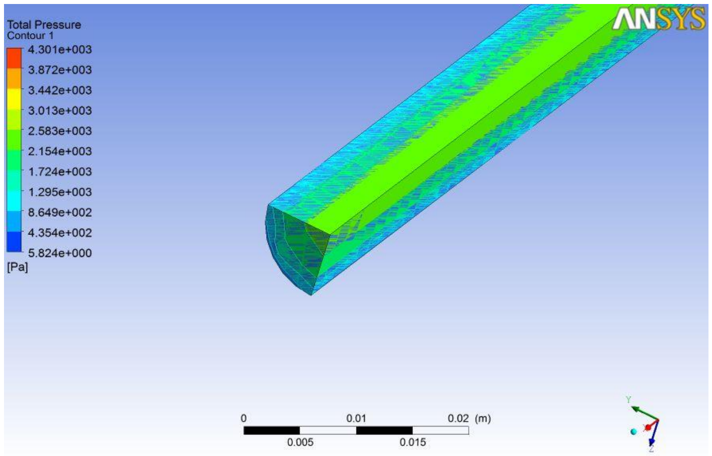


Fig 5.l. TOTAL PRESSURE CONTOUR FOR LIQUID AMMONIA

Maximum pressure observed is 4.3 KPa and minimum pressure 5.8 KPa. The inlet pressure is the maximum pressure observed.

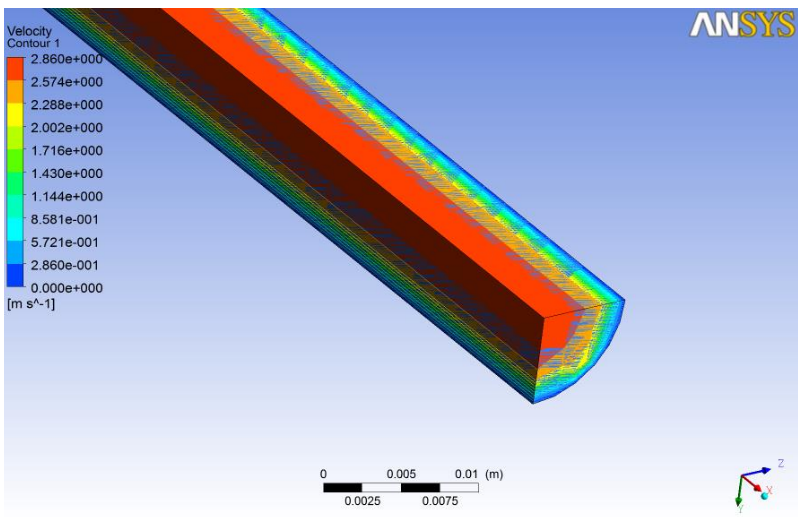


Fig 5.m. VELOCITY CONTOUR FOR DIESEL

As compared to liquid ammonia and water, the loss is more in case of diesel because of its high viscosity. For inlet velocity of 2.42 m/sec, maximum velocity is 2.86 m/sec and minimum is 0 at the walls.

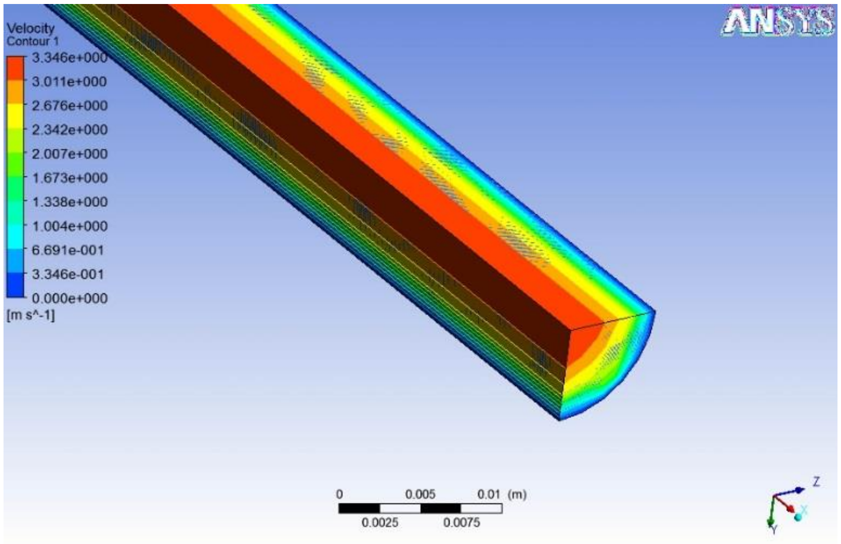


Fig 5.n. VELOCITY CONTOUR FOR CRUDE OIL

For an inlet velocity of 2.42 m/sec, the maximum velocity and minimum velocity were 3.34 m/sec and 0 respectively. Energy loss is maximum in case of crude oil flow.

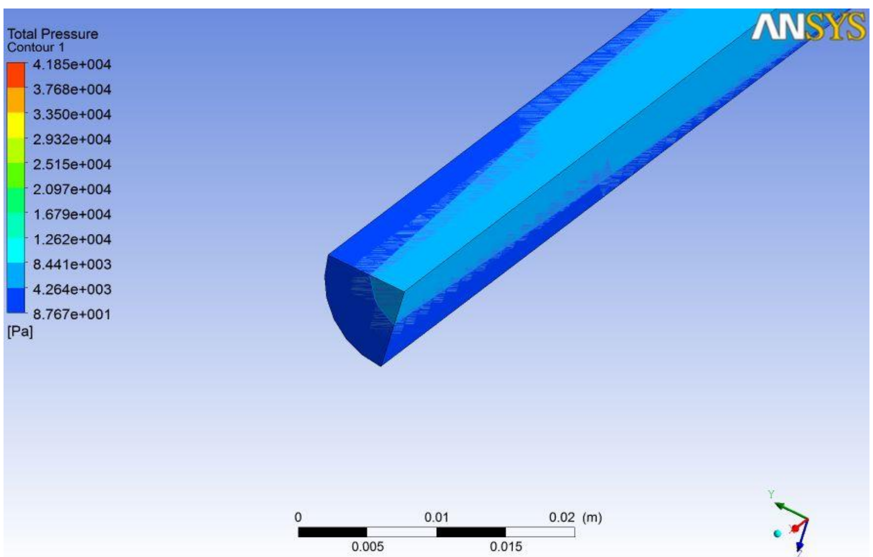


Fig 5.o. TOTAL PRESSURE CONTOUR FOR CRUDE OIL

5.4.3 STAINLESS STEEL PIPE

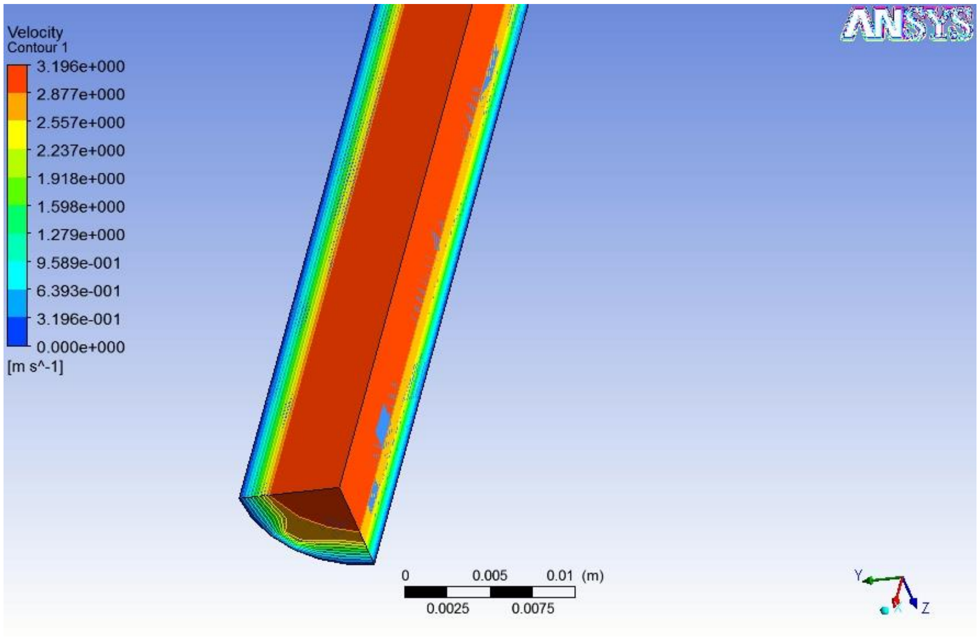


Fig 5.p. VELOCITY CONTOUR FOR LIQUID AMMONIA

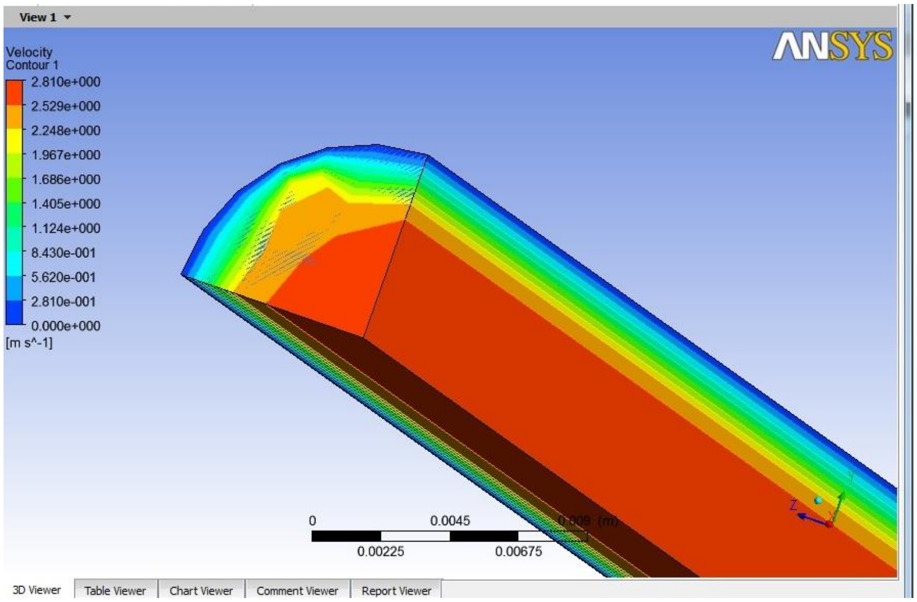


Fig 5.q. VELOCITY CONTOUR FOR WATER

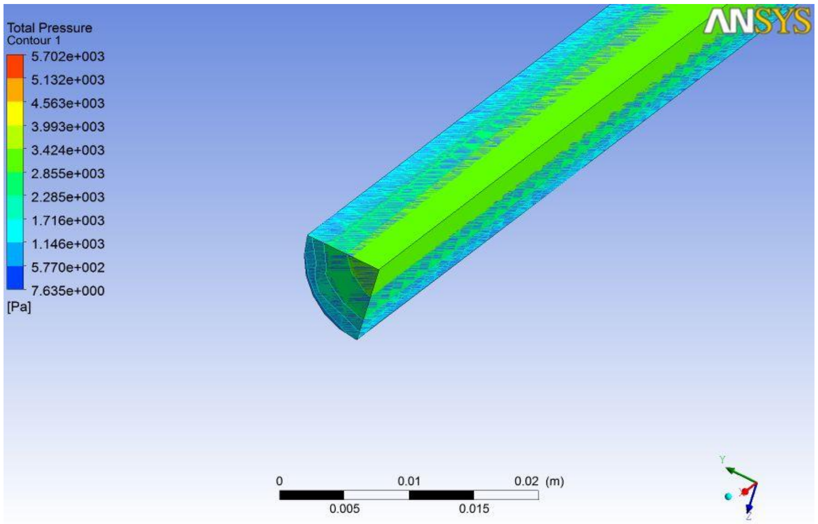


Fig 5.r. TOTAL PRESSURE CONTOUR FOR LIQUID AMMONIA

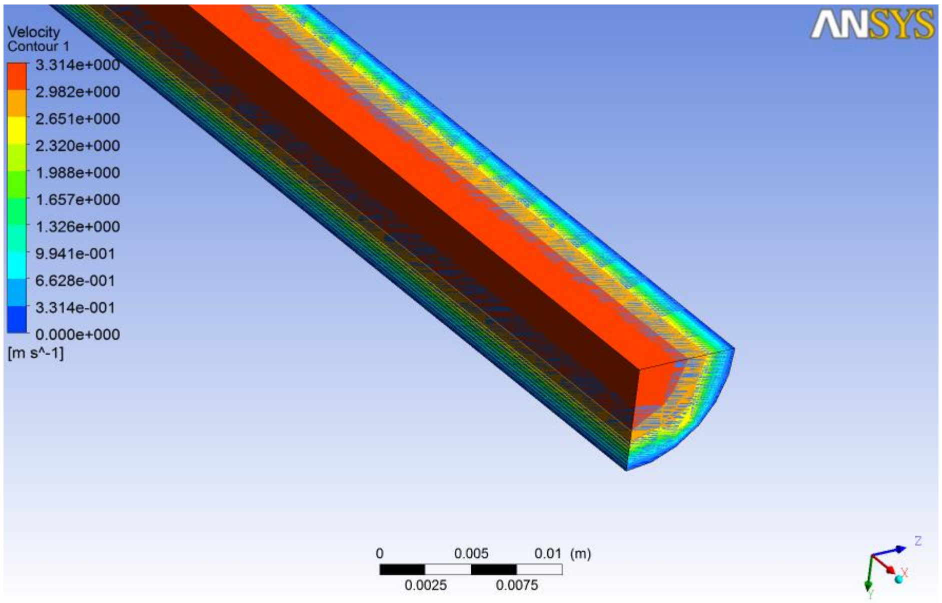


Fig. 5.s VELOCITY CONTOUR FOR DIESEL

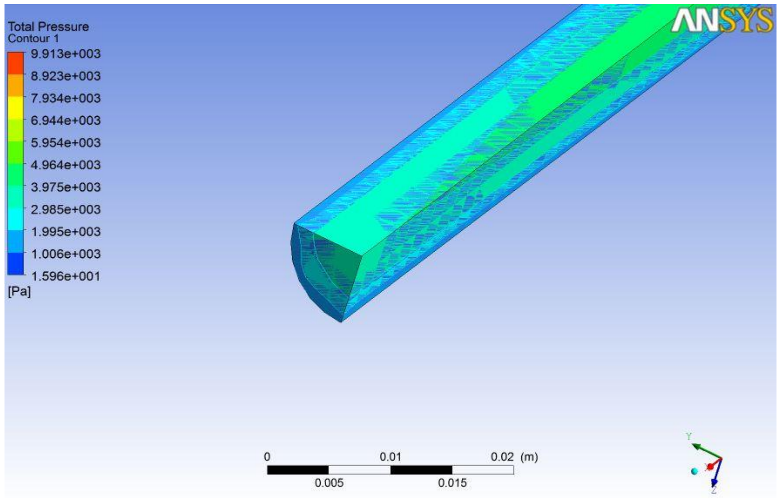


Fig 5.t. TOTAL PRESSURE CONTOUR FOR DIESEL

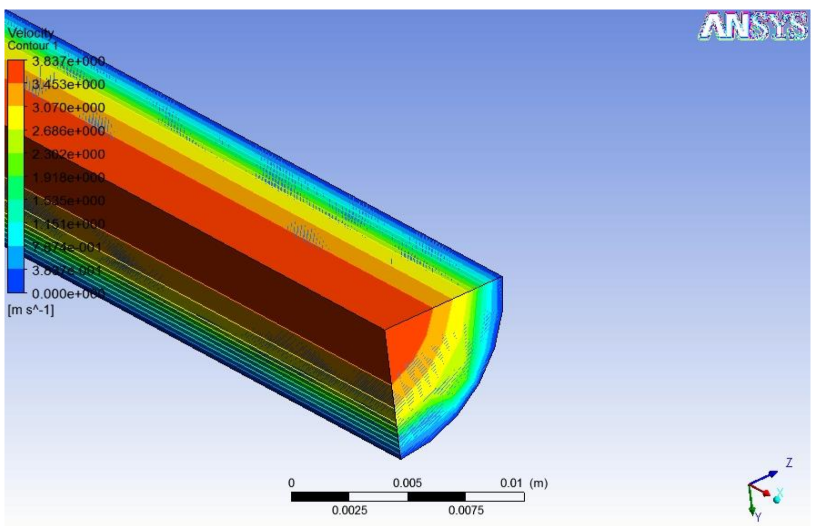


Fig 5.u. VELOCITY CONTOUR FOR CRUDE OIL

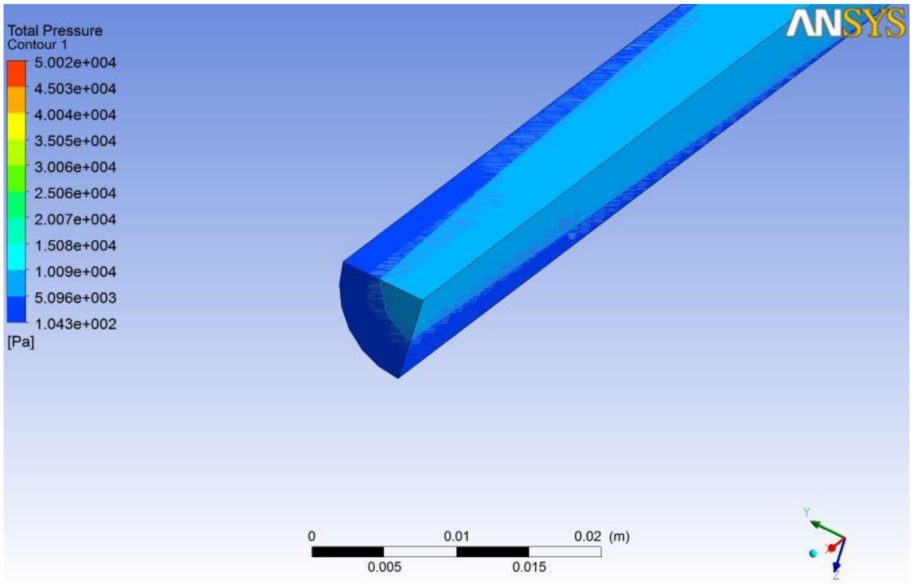


Fig 5.v. TOTAL PRESSURE CONTOUR FOR CRUDE OIL

5.5 FLOW ANALYSIS THROUGH A 25.4mm DIA PIPE

For minimizing frictional loss, diameter for a pipe can be increased. Earlier, analysis was done using a 12.7mm pipe. To check if loss decreases with increase in diameter of pipe, 25.4 mm diameter pipe with 1 m length was modelled. Water was allowed to flow through it and frictional losses were calculated using ANSYS.

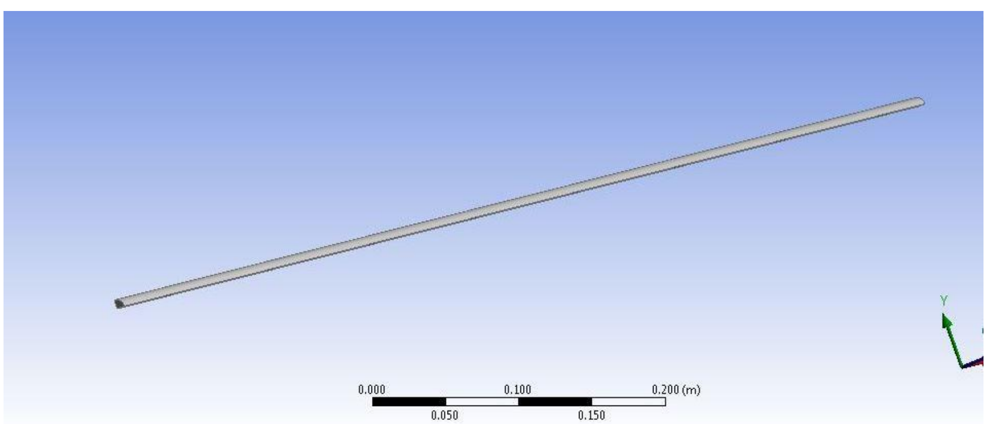


Fig 5.w. ANSYS MODEL OF A 25.4 mm PIPE

The pressure difference between the inlet and outlet gave the frictional loss. Below figures illustrate the velocity and total pressure contours for the pipe with an intake velocity of 2.56 m/sec.

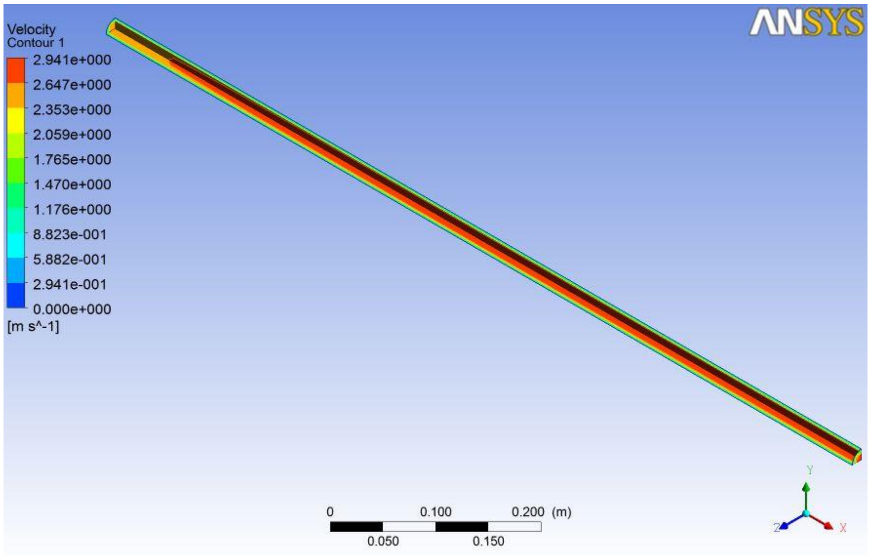


Fig 5.x. VELOCITY CONTOUR

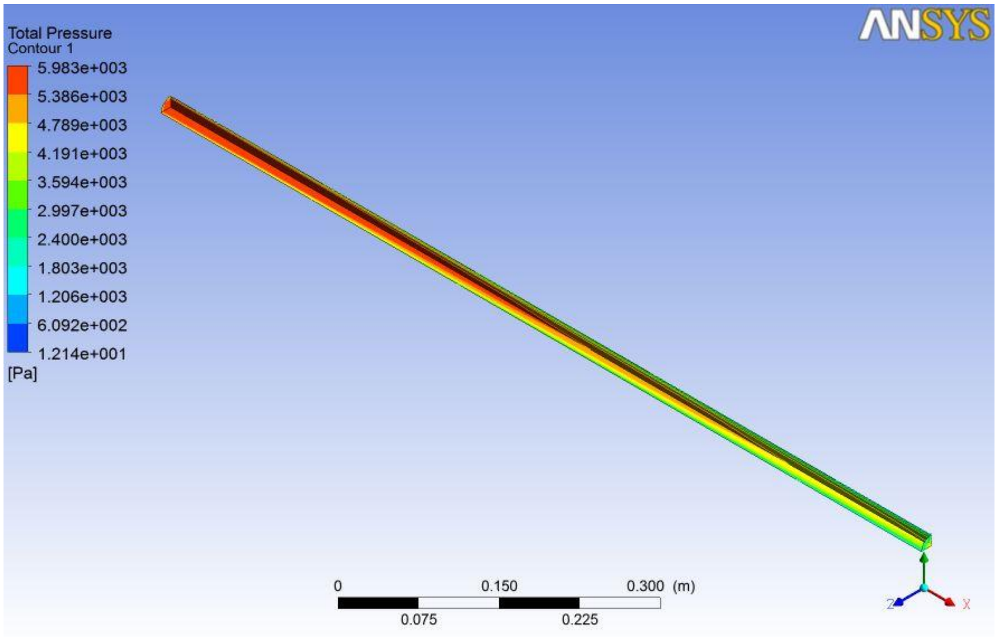


Fig 5.y. TOTAL PRESSURE CONTOUR

5.6 COMPARISON

Results obtained from ANSYS are compared below in a tabular form. The frictional coefficient is observed to lower in case of 25.4 mm dia pipe indicating that the losses are less when pipe diameter is more.

Table 5.2 (COMPARISON BETWEEN 12.7mm dia AND 25.4mm dia PIPE)

DIA OF PIPE (mm)	MATERIAL	FRICTIONAL COEFFICIENT
12.7	GALVANIZED IRON	0.024
25.4	GALVANIZED IRON	0.019
12.7	STAINLESS STEEL	0.02
25.4	STAINLESS STEEL	0.018
12.7	BRASS	0.023
25.4	BRASS	0.02

CHAPTER 6

RESULTS

AND

CONCLUSION

6.1 RESULTS

The results obtained are as follows.

1. PRACTICAL RESULTS (FLUID – WATER)

Friction coefficient for GI pipe = 0.024

Friction coefficient for Brass pipe = 0.023

Friction coefficient for Stainless Steel pipe = 0.02

2. THEORETICAL RESULTS (FLUID – WATER)

Friction coefficient for GI pipe = 0.025

Friction coefficient for Brass pipe = 0.024

Friction coefficient for Stainless Steel pipe = 0.022

3. ANSYS RESULT

FLUID – WATER

Friction coefficient for GI pipe = 0.021

Friction coefficient for Brass pipe = 0.021

Friction coefficient for Stainless Steel pipe = 0.02

FLUID – LIQUID AMMONIA

Friction coefficient for GI pipe = 0.0098

Friction coefficient for Brass pipe = 0.0097

Friction coefficient for Stainless Steel pipe = 0.0092

FLUID – DIESEL

Friction coefficient for GI pipe = 0.028

Friction coefficient for Brass pipe = 0.0278

Friction coefficient for Stainless Steel pipe = 0.026

FLUID – CRUDE OIL

Friction coefficient for GI pipe = 0.121

Friction coefficient for Brass pipe = 0.14

Friction coefficient for Stainless Steel pipe = 0.114

FLUID – WATER DIAMETER = 25.4 mm

Friction coefficient for GI pipe = 0.019

Friction coefficient for Brass pipe = 0.02

Friction coefficient for Stainless Steel pipe = 0.018

6.2 CONCLUSION

The study shows that the more viscous the fluid more is the frictional coefficient and thus higher the frictional loss. Thus, crude oil gives the experiences the maximum loss, followed by diesel, followed by water and then liquid ammonia. Frictional loss majorly depends upon the viscosity of the fluid than the pipe material. Loss observed is maximum for a G.I pipe, followed by Brass pipe and minimum in Stainless Steel pipe. Also, it is observed that increase in velocity decreases the frictional loss as the shear generated between wall and the fluid is less.

Increasing the diameter denotes a decrease in the frictional coefficient. When a 1 inch pipe (25.4 mm dia) was analyzed, it experienced less frictional loss than a ½ inch pipe (12.7mm dia).

REFERENCES

1. Bansal R.K. Fluid mechanics and hydraulic machines. Laxmi publication (p) ltd,(2010).

2. Abdulwahhaba Mohammed, Injetib N K, Dakhilc Sadoun Fahad “CFD Simulation and flow analysis through a T junction pipe” IJEST, 4(7),3393-3407,(2012).

3. Abdulwahhaba Mohammed, Injetib N K, Dakhilc Sadoun Fahad, “Numerical prediction of pressure loss of fluid in a T junction” IJEE, 4(2), 253-264,(2013).

4. Singh Navtej, Bansal Suneev Anil and Batra N.K. Visualization of flow behaviour and its effected contour in sudden contraction, sudden enlargement and elbow by ANSYS. IJREAS, l2 (5), 89-98.(2012).

5. Richards Scott, Martin Keith, and Cimbala John M. Minor Losses using Point wise and FLUENT. Penn State University, (2011).